MODELLING OF HIGH EFFICIENCY MINIATURE AEROCYCLONE

AIMAN HAKIM BIN BADARISMAN

A thesis submitted in fulfillment of the
requirements for the award of the degree of
Bachelor of Chemical Engineering

Faculty of Chemical and Natural Resources Engineering
Universiti Malaysia Pahang

APRIL 2010

ABSTRACT

Cyclone separation is one of the largely recognized methods of removing particulate in air, gas or water stream without help from filters. Cyclone is favourite separation device in various industries due to its simplicity, inexpensive design and minimum operational cost. In this research, the objective is maximizing the performance of miniature aerocyclone. The performance of aerocyclone can be analyzed by its collection efficiency and pressure drop. In past studies, expensive and time consuming experiment were conducted in order to analyze the performance of cyclone, but Computational Fluid Dynamics (CFD) software that being implemented in this research has successfully solve the problem. FLUENT is a commercially available CFD code that utilizes the finite volume formulation to carry out coupled or segregated calculations. For the turbulent flow in a cyclone the key to the success of CFD lies with the accurate description of the turbulent behavior of the flow. The simulation methodologies involve 5 steps before the simulation data can be verified with experimental data. The simulation starts with meshing of miniature cyclone via Gambit software. The meshed cyclone was then transferred into FLUENT software for simulation, where all the boundary condition is programmed in the software. After simulation completed, the data was then extracted and converted in the function of cyclone collection efficiency and pressure drop. The results show the influence of inlet velocity and exit tube size on cyclone performance and significant conclusion were made. The performances of turbulence model used in the simulation were then compared.

.

ABSTRAK

Pemisahan menggunakan alat siklon adalah salah satu kaedah yang diakui sebahagian besar sangat berkesan mengasing partikel di udara, gas, atau air sungai tanpa bantuan daripada penapis. Siklon adalah peranti pemisahan lazim di pelbagai industri kerana kesederhanaan rekaan dan kos operasi yang minimum. Objektif utama kajian ini adalah memaksimumkan prestasi mini siklon udara menggunakan teknologi Computational Fluid Dynamics (CFD). Prestasi asiklon udara boleh dianalisis dari kecekapan pungutan partikel dan perbezaan tekanan didalam siklon. Daripada kajian silam, kaedah eksperimen ternyata mahal dan memakan masa dalam menganalisis prestasi siklon. Perisian Computational Fluids Dynamics (CFD) yang diimplementasikan di dalam kajian ini telah berjaya mengatasi masalah tersebut. FLUENT adalah kod CFD sedia ada komersil yang memanfaatkan formulasi tentu isipadu dalam melaksanakan pengiraan secara pasangan dan mahupun terpisah. Kunci utama ketepatan analisa CFD terhadap aliran turbulen bergantung pada input data yang tepat untuk perilaku turbulen. Metodologi simulasi melibatkan 5 langkah sebelum data simulasi boleh disahkan dengan data eksperimen. Simulasi bermula dengan kaedah “meshing” model maya mini siklon menggunakan perisian Gambit. Siklon yang telah di’mesh’ kemudian dipindahkan ke dalam software FLUENT untuk simulasi, di mana semua keadaan had diprogramkan dalam perisian FLUENT. Setelah selesai simulasi, data kemudian diekstrak dan diterjemah dalam fungsi kecekapan pengumpulan dan perbezaan tekanan didalam siklon. Prediksi CFD menunjukkan kelajuan masuk udara dan dimensi tiub keluar siklon amat mempengaruhi prestasi siklon, sesuai dengan data eksperimen. Model turbulen (RSM & DES) yang digunakan dalam simulasi kemudian dibandingkan prestasinya.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

L	-	Natural length (m)
a	-	cyclone inlet height (m)
b	-	cyclone inlet width (m)
D	-	cyclone body diameter (m)
De	-	cyclone gas outlet diameter (m)
H	-	cyclone height (m)
h	-	cyclone cylinder height (m)
μ	-	Viscosity of liquid (Pa.s)
h	-	Heat transfer coefficient
^oC	-	Degree Celsius
kg	-	Kilogram
K	-	Degree Kelvin
m	-	Meter
S	-	cyclone gas outlet duct length (m)
B	-	cyclone dust outlet diameter (m)
d_p	-	particle diameter (m)
v_i	-	inlet velocity (m/s)
C_D	-	Drag Coefficients

CHAPTER 1

INTRODUCTION

1.0 Introduction

Cyclone separation is one of the largely recognized methods of removing particulate in air, gas or water stream without help from filter. These particulates are removed through spinning, often turbulent, flow of fluid plus a little help from gravity. The device is called a cyclone. A conventional cyclone looks like a cylindrical or conical where high velocity air rotating within it. At the top of the cyclone (wide end) air flow as vortex and end up at the narrow end or the bottom of the cyclone. The air flow then goes out through top in a straight stream via center of the cyclone, meanwhile denser particle in the rotating stream have too much inertia to follow the tight curve of the stream and strike the outside wall, falling then to the bottom of the cyclone where they can be removed. Within the conical section of the cyclone the rotational radius of stream is reduced whenever the flow move nearer to the narrow bottom end, thus separating smaller and smaller particles. The outlet flow at the top end often labeled as major flow meanwhile the outlet flow at bottom where dust container is placed often labeled as minor flow. A cyclone where the minor flow is pumped out from its container is called virtual cyclones.

Compared to other gas–solid separation methods, such as filters, scrubbers, and settlers, cyclone separators are simpler to construct, easier to maintain, require lower capital and operating costs, are more compact, and versatile. Especially at high temperatures and pressures, the cyclone is often the only viable economical separation process (Wang et al., 2010). Cyclones low capital cost and seldom maintenance operation make them suitable for use as pre-cleaners for more expensive final control devices such as bag-houses or electrostatic precipitators. Cyclone can also be beneficial to human health. For examples, in a saw-mill or a power plant, large scale cyclone is use to removed sawdust or coal dust from the air. Both sawdust and coal dust are dangerous when enters human respiratory system. In oil refinery industries, large scale cyclone is used to separate oil and gases meanwhile in cement industries; cyclone is implemented as one of the component of kiln pre-heaters.

Large cyclones are used to remove particles from industrial gas streams, while small cyclones (see figure 1.1) are used to separate particles for ambient sampling (Kim et al., 2001). With mini cyclone, fine particle as small as dust can be separated from air. Due to its size, mini cyclones can be located in various places, making it suitable for analysis instruments. Small cyclones are in the use of the so-called personal cyclones for the selective sampling of various air-laden respirable particles in occupational hygiene and environment control. The cyclones which are used in these special circumstances are usually very small in size with the typical working flow rate of a personal cyclone being about 2 liter/min in order to sample particles of just a few microns in diameter. Personal cyclones used in occupational hygiene and environment hygiene often use light battery-operated pumps which are available with a flow rate up to 4-5 liter/min.

Figure 1.1: Miniature Aero-cyclone

When engineers want to design a cyclone, there are two important parameters that need to be considered. These parameters are collection efficiency of particle and pressure drop across the cyclone. An accurate prediction of cyclone pressure drop is very important because it relates directly to operating costs. Higher inlet velocities give higher collection efficiencies for a given cyclone, but this also increases the pressure drop across the cyclone (Gimbun et al. 2004). Particle size distribution is also strongly influence the collection efficiency. Cyclone collection efficiencies can reach 99% for particles bigger than 5 μm (Silva et al. 2003). Their simple design, low maintenance costs, and adaptability to a wide range of operating conditions such as sizes and flow rates make cyclones one of the most widely used particle removal devices. By using suitable materials and methods of construction, cyclones may be adapted for use in extreme operating conditions: high temperature, high pressure, and corrosive gases (Gimbun et al. 2004).

1.1 Computational Fluid Dynamics (CFD)

Miniature cyclone will be modeled using Computational Fluid Dynamics (CFD). CFD is a computational technology that enables study of the dynamics of things that flow. Using CFD, one can build a computational model that represents a system or device that is want to studied. The fluid flow physics and chemistry is applied to this virtual prototype, and the software will output a prediction of the fluid dynamics and related physical phenomena. Therefore, CFD is a sophisticated computationally-based design and analysis technique. CFD software gives the capabilities to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling. Using CFD software, one can build a 'virtual prototype' of the system or device that one wish to analyze and then apply real-world physics and chemistry to the model, and the software will provide images and data, which predict the performance of that design.

The CFD solver does the flow calculations and produces the results. Four general-purpose products: FLUENT, FloWizard, FIDAP, and POLYFLOW. FLUENT is used in most industries. FloWizard is the first general-purpose rapid flow modeling tool for design and process engineers built by Fluent. POLYFLOW (and FIDAP) are also used in a wide range of fields, with emphasis on the materials processing industries.

The FLUENT CFD code has extensive interactivity; changes can be made to the analysis at any time during the process. These saves time and enable design refines more efficiently. The graphical user interface (GUI) is intuitive, which helps to shorten the learning curve and make the modeling process faster. It is also easy to customize physics and interface functions to specific needs. In addition, FLUENT's adaptive and dynamic mesh capability is unique among CFD vendors and works with a wide range of physical models. This capability makes it possible and simple to model complex moving objects in relation to flow.

FLUENT as a commercially available CFD code utilizes the finite volume formulation to carry out coupled or segregated calculations (with reference to the conservation of mass, momentum and energy equations). It is suitable for incompressible to mildly compressible flows. The conservation of mass, momentum and energy in a fluid flow are expressed in terms of non-linear partial differential equations that generally defy solution by analytical means. The solution of these equations has been made possible by the advent of powerful workstations, as available in Universiti Malaysia Pahang (UMP) Process Control Lab, opening avenues towards the calculation of complicated flow fields with relative ease. Figure 1.2 shows the analysis of dynamic behaviour of various objects using CFD software.

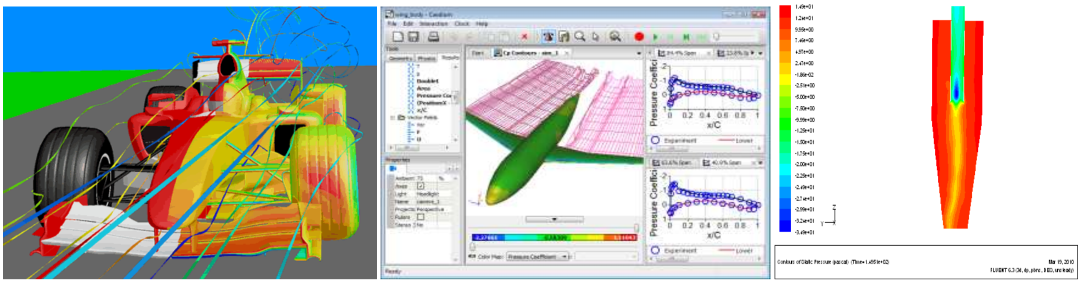

Figure 1.2: The applications of CFD in simulating dynamics behavior of F1 car, airplane and cyclone separation.

1.2 Research Objectives

The following are the objectives of this research:

1) To evaluates the influence of vortex finder dimension and inlet velocity on miniature aero-cyclone collection efficiency and pressure drop
2) Modeling high efficiency miniature aero-cyclone using Computational Fluids Dynamic (CFD)

1.3 Problem Statement

The pressure drops and collection efficiency are evidently the most important features from the point of view of cyclonic operation. At the end of this research we want to see the relationship of these two features with the design and operating parameters, or the manipulated parameters. Manipulated parameters for this research are vortex finder dimension and inlet velocity. In the past, the only way to determine the collection efficiency and pressure drop of cyclone operation is via experiment. Let say we want to distinguish the effect of two cyclones with different tip diameters towards pressure drop; experimentally we have to build the prototypes of these two different cyclones. These prototypes then operated and pressure across the cyclone is measured. Finally the data is compared between the two cyclones before conclusions were drawn. These works require plenty of time and investment moreover if we have many design parameters to be tested. Online Master-sizer and LDA are the instruments used to determine the cyclone collection efficiency and pressure drop experimentally, and they are not come cheap. This is where simulation technology comes in handy. Over the year, computer and software technology has advances so much where the effect of design parameter on cyclone can now be simulated with one simple click. Thus, CFD is fully utilized in this study to understand the behavior of cyclone separation.

CHAPTER 2

LITERATURE REVIEW

2.0 Introduction

With growing concern for the environmental effects of particulate pollution, it is becoming increasingly important to be able to optimize the design of pollution control systems; the separation of solid particles is also required in many industrial processes. For this purpose, cyclone separators are widely used as the most common devices. Conventionally, cyclone separators have been used as pre-cleaning devices for the removal of particles bigger than 10 μm from the carrier gas in both air pollution control and other processes. Because of their adaptability, simple design and low costs in terms of maintenance, construction and operation, those properties make cyclones ideal for use in the various stages of industrial applications.

Due to cyclone effectiveness in industry, numerous researches were conducted related to influence of cone tip diameter, particle size distribution, vortex finder dimension, inlet velocity, cyclone height and operating temperature on miniature aero-cyclone collection efficiency and pressure drop. In the last few decades, the majority of the research on cyclones has focused on large industrial cyclones and a number of predictive models have been developed which attempt to predict key performance parameters, such as the characteristics of the separation efficiency and the pressure drop across the cyclone. The need to improve the collection efficiency of a cyclone has existed since the early 1950, where Linden (1949), Lapple (1950), and Stairmand (1952) did their research on particulate removal from contaminated gas streams in industrial processes. Pressure drops across the cyclone plays a great deal in determining the performance of cyclone, where usually researchers conclude the pressure drop change along with collection efficiency, for example Kim & Lee (1990), Griffith & Boysan (1996), Kim et al. (2001), and Chuah et al. (2006).

Miniature cyclone was comprehensively studied when industrial hygiene awareness significantly increases in the public eye. In the case of health-related personal cyclones, more strict standards are enforced on industrial hygiene than is the situation for cyclones which are used in industrial processes and this has been outlined by Kenny (1996). In order to model the behaviour of the human breathing, personal cyclones are required to possess a particle size-selection (or penetration/separation) curve which follows specific sampling convention, for example the ISO international sampling convention (Ma et al., 2000). In recent years, a great deal of effort has continuously been contributed to the research on small sampling cyclones, for example studies made by Chan and Lippmann, 1977; Liden and Kenny, 1991, 1992; Overcamp and Scarlett, 1993; Maynard and Kenny, 1995; Kenny and Gussman, 1997. It is found that most of the existing predictive models for cyclones are experimentally based semi-empirical models and do not have a profound fluid mechanics basis. Therefore, they are often limited in their use to some special type of cyclones. The design of new cyclones for specific applications often rely on performing numerous and very costly experimental investigations. It is not until recently that relatively rapid and accurate methods have become available to measure the particle size-selective characteristics of small personal cyclones.

2.1 Studies on Cyclone Performance

2.1.1 Effect of Vortex Finder and Cone Dimension on Cyclone Performance

Vortex finder can also be referred to the exit tube of the cyclone which is fixed on the top of the cyclone and itprotrudes some distance into the body of the cyclone (refer figure). The vortex finder size is an especially important dimension, which significantly affect the cyclone performance as its size plays a critical role in designing the flow field inside the cyclone, including the pattern of the outer and inner spiral flows. Saltzman and Hochstrasser (1983) studied the design and performance of miniature cyclones for respirable aerosol sampling, each with a different combination of three cyclone cone lengths and three vortex founder diameters. Iozia and Leith (1989) optimized the cyclone design parameters, including the vortex founder diameter, to improve the cyclone performance using their optimization program. Lim et al. (2003) found that the collection efficiencies of the cyclones with the cone-shaped vortex finders varied slightly, thus they conclude that cone shapes or lengths of cone-shaped vortex finder are not an important factor in affecting the particle collection efficiencies. Meanwhile Kim & Lee (1990) believes that the exit tube size plays a critical role in defining the flow field inside the cyclone, including the pattern of the outer and inner spiral flows.

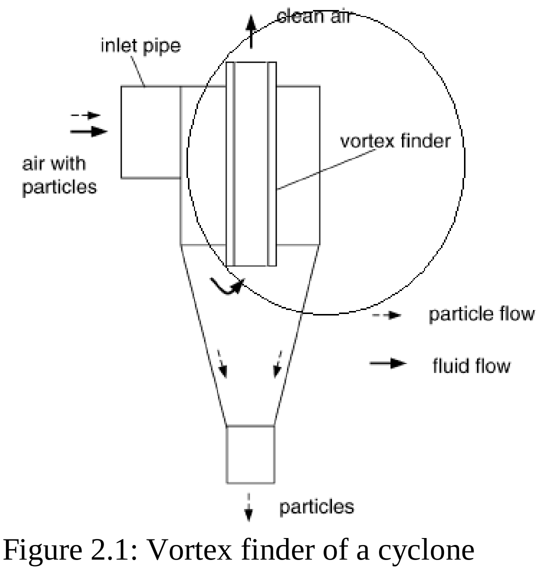

Figure 2.1: Vortex finder of a cyclone

The cone dimension of a cyclone is actually influential on cyclone performance even though it was usually neglected in many cyclone body dimension researches. The cone was considered only for practical purpose of delivering collected particles into bottom discharge point. Chuah et al. (2005) proves cyclone with a smaller cone diameter result slightly higher collection efficiency compared to the cyclone with a bigger cone diameter using CFD simulation. The simulation were done using two turbulence model, RNG k-ε and RSM based on Xiang et al. (2001) experiment.

2.1.2 Effect of Inlet Velocity on Cyclone Performance

Inlet velocity of mixture between gas and particle was proven influential to cyclone performance. Inlet velocity can be obtained by dividing the volume flow rate of inlet with inlet area of the cyclone vane. Gimbun et al. (2004) stated that for the identical size and configuration of cyclone, the higher the gas inlet velocity is, the sharper the efficiency would be. However, a very high inlet velocity would decrease the collection efficiency because of increased turbulence and saltation/re-entrainment of particles. Kim & Lee (1990) reported that the collection efficiency is seen to increase as either the particle size or the flow rate increases, where they applied three different inlet velocities which are 8.8, 12.4 and 18.4 liter/min.

Pressure drop prediction is important as it will influence the operating cost of cyclones. As Chuah et al. (2003) cited in their report that higher inlet velocities give higher efficiencies for a given cyclone, but this also increases the pressure drop, and a trade-off must be made. They had calculated the pressure drop using 4 different empirical models such as Shepherd and Lapple (1939), Casal and Martinez (1983), Dirgo (1985), and Coker (1993) model and subsequently compare the result to experimental value made by Patterson and Munz, (1989) for the prediction under different inlet velocity.

2.1.3 Effect of Temperature on Cyclone Performance

Effect of temperature on collection efficiency of cyclone is one of the most essential researches. Chen & Shi (2005) analyze this factor, where an experimental investigation on particle separation was conducted in a 300 mm diameter, tangential volute-inlet and reverse flow cyclone separator with air heated up to 973 K. They presents the experimental results on fractional as well as overall collection efficiencies of a test cyclone, aiming at better understanding the separation mechanism and improving the prediction of collection efficiencies at high temperatures which they found that at the same inlet velocity both the overall efficiency and fractional efficiency decrease with an increase of temperature.

Bohnet (1995) has measured pressure drop and the grade efficiency over the temperature range 293-1123 K since practical experience shows that the separation efficiency really obtained is mostly smaller than that calculated. A new model into which a wall friction coefficient depending on the temperature is introduced and consideration taken of the re-entrainment of already separated particles together with the boundary layer flow at the cyclone cover plate, the cyclone wall and along the outside of the vortex finder leads to measured pressure drops and grade efficiency curves for high temperatures of a high accuracy. He concluded similar to Chen & Shi where when temperature is high, the collection efficiency decrease, so as pressure drop.

2.1.4 50% Particle Cut-Off Diameter

Cut-off size of particle diameter can be defined as particle size at 50% cyclone collection efficiency. The particle cut-off size d_{50} of the cyclone are always employed in the analysis of cyclone performance because its often give simple and clear indication of the particle collection efficiency of the cyclone and this makes it easy to compare the performances of cyclones under different operating conditions or of different geometrical designs. Previous experimental studies have showed that an increase in the inflow rate results in a decrease of cut size diameter (Moore & McFarland,1990; Iozia & Leith, 1990; Zhu & Lee, 1999; and Kim & Lee, 1990). Kim & Lee (1990) observed that d_{50} is inversely proportional to inlet velocity of cyclone. As the inlet flow rate increase the collection efficiency curve become sharper, means smaller $d_{50.}$ According to them, this is because at given flow rate it is possible to alter particle cut size diameter without deteriorating the sharpness simply by adjusting the exit tube size.

Kim et al. (2001) whose studied on virtual cyclone (minor flow pumped out at the bottom) found that compared to conventional cyclone, virtual cyclone showed smaller 50% cut-off diameter under the same operational condition. Subsequently, they correlate that pressure drop is more dependent to inlet flow compared to $d_{50,}$ Chuah et al. (2005) made a CFD prediction of d_{50} using two different turbulence model, k-ε and RSM, and they demonstrated that RSM turbulence model is more accurate method of modeling the 50% cut off diameter with average deviation of about 2.1% of the measured value compared to 4.9% of RNG k-ε.

2.1.5 Review on Experimental and Simulation Studies on Cyclone Performance

The need to enhance cyclone performance has surged since the early 40's. Due to limitation of technology, scientist need to built cyclone prototype in order to understand the influential parameters toward cyclone performance, which prove costly and time consuming. Nowadays, computer and software technology bloom enables researchers to simulate the dynamic model of cyclone, for example using Computational Fluid Dynamic (CFD) software. Table 2.1 show the summary of experimental and simulation study made by several researchers.

Nu.	Author	Modelling CFD					Experiments					Remarks
		Turbulence	Temperature	Collection Efficiency	Pressure Differen-ce	Particle Size Distribution	Turbulence	Temperature	Collection Efficiency	Pressure Difference	Particle Size Distribution	
1	Chuah et al. (2006)	Yes Renault Stress Model (RSM), RNG K-ε	No	Yes	Yes	Yes	No	No	No	No	No	RSM turbulence model predict cyclone collection efficiency better than RNG k- ε. Smaller cone diameter result in higher collection efficiency
2	Bohnet (1995)	No	No	No	No	No	No	Yes	No	Yes	Yes Light Scattering	High temperature separation efficiency smaller. High temperature, small pressure difference.
3	Altmeyer et al.(2004)	No	Yes Leith and Licht model	Yes	Yes Mothes and Löffler model	Yes Mothes and Löffler model	No	No	No	No	No	Studies of literature cases show that models used predict pretty well the experimental results.

Nu.	Author	Modelling CFD					Experiments					Remarks
		Turbulence	Temperature	Collection Efficiency	Pressure Differen-ce	Particle Size Distribution	Turbulence	Temperature	Collection Efficiency	Pressure Difference	Particle Size Distribution	
4	Fassani et al. (2000)	No	No	No	No	No	No	No	Yes Function of solid loading	Yes Function of solid loading	No	The introduction of solid particles in the gas flow brought about a reduction in the cyclone pressure drop Pressure drop essentially did not vary as a function of solid loading Increasing separation efficiency With high concentration of solid loading
5	Chen et al. (2003)	No	No	No	No	No	No	Yes	Yes Function of temperatu-re	No	Yes	The overall collection efficiencies of a cyclone decreases with increase in temperatures. The critical particle size below which the fractional efficiency increases as the particle size decreases
6	Reijnen et al. (1984)	No	No	No	No	No	No	Yes HTHP Gas Cleaning	Yes	Yes HTHP Gas Cleaning	No	Review of gas cleaning method to be used as particulate remover

Nu.	Author	Modelling CFD					Experiments					Remarks
		Turbulence	Temperature	Collection Efficiency	Pressure Differen-ce	Particle Size Distribution	Turbulence	Temperature	Collection Efficiency	Pressure Difference	Particle Size Distribution	
7	Wang et al. (2005)	No	No	No	No	No	Yes Laser Doppler Velocime-try (LDV)	No	Yes	Yes Laser Doppler Velocim e-try (LDV)	No	Insertion of a stick into the cyclone separator forms a flow wake region that decreases the tangential velocity, and increases the RMS velocity and turbulence intensity.
8	Obermair et al. (2003)	No	No	No	No	No	Yes LDA velocity measure-ments	No	Yes	No	No	LDA velocity measurements of the clean gas flow of a gas cyclone with three different dust outlet geometries show that both the flow in the dust outlet geometry and the flow within the lower part of the cyclone change with outlet geometry
9	Gimbun et al. (2005)	Yes Renault Stress Model (RSM), RNG k-ε	No	Yes	No	Yes Cut-off diameter prediction	No	No	No	No	No	The Li and Wang model and CFD code both predict very well the cyclone efficiency and cut-off size for any operational conditions
10	Kim and Lee (1990)	No	No	No	No	No	No	No	Yes	Yes Exit tube size effect	Yes 50% cut-off diameter in the function of exit tube size	Exit tube size effect cyclone performance significantly